

Wind Tunnel Model Design for Sonic Boom Studies of Nozzle Jet with Shock Interactions

Susan E. Cliff¹

Ames Research Center, Moffett Field, California, 94035

Marie Denison², Emre Sozer³, & Shayan Moini-Yekta⁴

Science and Technology Corporation, Moffett Field, California, 94035

NASA and Industry are performing vehicle studies of configurations with low sonic boom pressure signatures. The computational analyses of modern configuration designs have matured to the point where there is confidence in the prediction of the pressure signature from the front of the vehicle, but uncertainty in the aft signatures with often greater boundary layer effects and nozzle jet pressures. Wind tunnel testing at significantly lower Reynolds numbers than in flight and without inlet and nozzle jet pressures make it difficult to accurately assess the computational solutions of flight vehicles. A wind tunnel test in the NASA Ames 9- by 7-Foot Supersonic Wind Tunnel from Mach 1.6 to 2.0 will be used to assess the effects of shocks from components passing through nozzle jet plumes on the sonic boom pressure signature and provide datasets for comparison with CFD codes. A large number of high-fidelity numerical simulations of wind tunnel test models with a variety of shock generators that simulate horizontal tails and aft decks have been studied to provide suitable models for sonic boom pressure measurements using a minimally intrusive pressure rail in the wind tunnel. The computational results are presented and the evolution of candidate wind tunnel models is summarized and discussed in this paper.

I. Abstract

The Commercial Supersonic Technology Project under NASA's Fundamental Aeronautics Program is developing technologies to enable future supersonic civilian aircraft to fly over land with reduced sonic boom. NASA's industry partners have been developed designs without the effects of the nozzle plume and have recently begun including the inlet and nozzle jet plume in their sonic boom predictions. The effect of the inlet and nozzle jet flow is configuration dependent, dependent upon the tail size, position and shock strength of the component as it interacts with the nozzle plume.

An experiment in the 1x1-Foot Supersonic Wind Tunnel at GRC was performed in 2014 where an isolated analytically defined nozzle body was tested using a single static pressure probe that traversed through the pressure field on the nozzle jet with a simple upswept double-wedge shock generator and several aft deck shapes fabricated with a 3-D printer that slid over the nozzle body. CFD results were good when the wind tunnel model was included in the solutions, but the close proximity of the walls of the tunnel restricted the range and altitudes of the sonic boom data.

The wind tunnel test to be performed in February of 2016 in one of America's largest supersonic wind tunnel facilities is a follow-on test that should provide complete pressure signatures for altitudes of 7, 15, 25 and 35 inches below the nozzle centerline at Mach 1.6 and possibly larger Mach numbers to Mach 2.0. It is expected that measurements of the complete signature including fronting forebody shock, the nozzle exit shocks, the entire signature of all shock generators (leading and trailing shocks) after passing through the plume, and some portion of the pressure field beyond the trailing edge within the nozzle plume. The data is expected to be of high-quality from the developments in sonic boom testing in the N+2 studies and the use of averaged pressure signatures using the Reflection Factor 1 (RF1) pressure rail.

The Ames 9- by 7-Foot Supersonic Wind Tunnel is part of the Unitary Plan Facility at NASA Ames Research Center. It is a continuous flow, closed circuit tunnel equipped with an asymmetric sliding-block nozzle for setting

¹ Aerospace Engineer, Applied Modeling and Simulations Branch, NASA Ames, AIAA Associate Fellow

² Research Engineer, CST Corp at NASA Ames, AIAA member

³ Research Engineer, CST Corp at NASA Ames, AIAA member

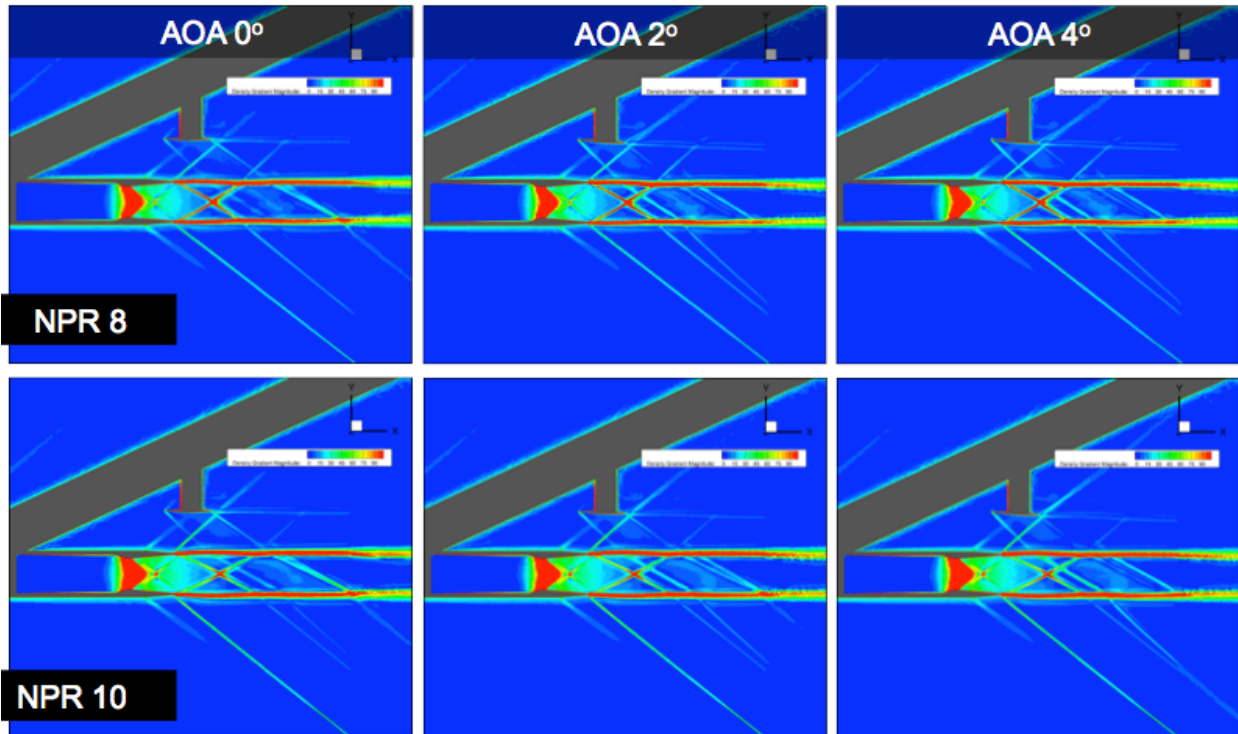
⁴ Research Engineer, CST Corp at NASA Ames, (no longer under contract)

the tunnel test section free-stream Mach number. The floor of the wind tunnel test section is the movable part of the nozzle block. It translates axially (streamwise) to vary the throat area and provides a Mach number range of 1.55 to 2.55. The stream angle varies throughout the test section by approximately 0.25° to 0.5° , and it is greater in the vertical plane because of the asymmetric nozzle formed by the floor and ceiling. Hence, models are mounted wings vertical in the test section because of this stream angle bias, and therefore the angle of attack plane is horizontal.

The RF1 pressure rail mounts to the north sidewall of the test section to measure the pressure signatures “below” the model. It has a 14” stand-off distance of the pressure orifices from the tunnel wall that provides reflection-free data for a model up to 35 inches long at Mach 1.6, without contamination from shocks reflected from the wall. The distance from the model to the tip of the rail is controlled by the wind tunnel strut, which moves the model horizontally in the test section. The strut is located at the down stream end of the test section and its center body contains a mechanical joint to control the angle of attack of the model and associated mounting hardware. The tunnel can operate at total pressures between 634 and 3600 psf with corresponding Reynolds numbers between 0.5 and 5.7 million per foot. For sonic boom tests, the Mach number and total pressure are prescribed rather than Reynolds number, and for sonic boom tests the tunnel is set at a P_T of 2300 psf (slightly above atmospheric pressure). This provides a Reynolds number per foot of 4.43×10^6 .

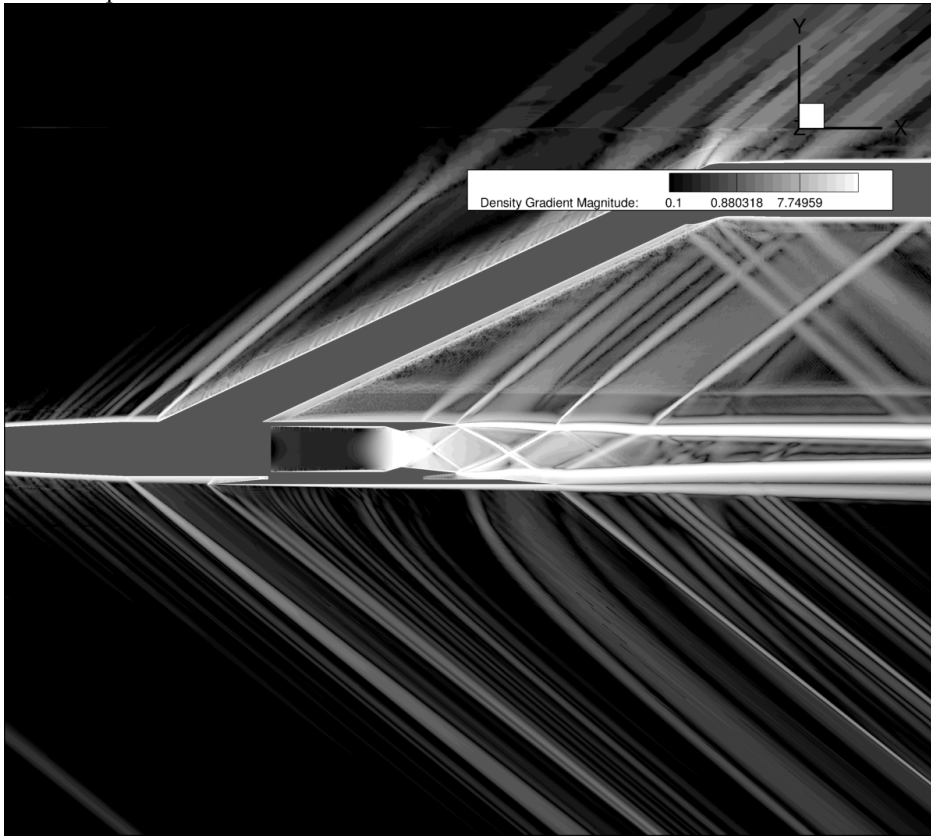
Design configurations could be generated relatively quickly by using a script for generating parameterized geometry with structured meshes using Chimera Grid Tools (CGT) software developed for overset methods to generate primarily analytically-defined surface meshes. These surface meshes were then used for viscous unstructured grids suitable for Navier-Stokes solutions via a semi-automated process utilizing StarCCM+ volume grid generation. These intermediate design solutions were obtained with the Launch Abort Vehicle Analysis (LAVA) code. Studies of fore-body shape, strut sweep, nozzle diameter, nozzle pressure ratios, and size and positions of shock generators (SG) were obtained and some of these studies will be presented in the paper. The designs were handed in STL (Stereo Lithography) format to a model designer for CAD based designs of the models for the experiment’s Preliminary Design Review (PDR).

The solutions of the configurations before PDR (Pre-PDR) will be presented at Mach 1.6. For example, a 1.5 inch root chord swept shock generator with a biconvex sections at NPR of 8 and 10 are shown with the SG at 0, 2, and 4 degrees angle of attack in the LAVA solutions in the figure. This biconvex SG is planned to be run at 0 and 4-degrees angles of attack in the upcoming experiment (separate parts will be manufactured for each angle of attack).



In addition to “model tail” type shock generators, we will be assessing the effects that aft decks have on the nozzle plume. For example, the symmetry plane of the flow solution with an aft deck that extends 1.8 nozzle diameters from the nozzle exit is shown at Mach 1.6 at a nozzle pressure ratio of 8 in the figure below. The

magnitude of the density gradient is similar to what should be visible with the planned Schlieren photography for the experiment. The upward deflection of the nozzle plume is seen in the CFD solution below and this behavior will be evaluated in the upcoming experiment using Schlieren and a pressure rake that will measure the total pressure at several axial positions behind the nozzle.



Solutions from all the CAD model PDR models and revised strut models will be presented in the final paper. Sonic boom pressure signatures and density gradients solution images of all these cases will be presented at Mach 1.6 before the experiment.